

RFlex Crankshaft Tutorial (Durability)





Copyright © 2020 FunctionBay, Inc. All rights reserved.

User and training documentation from FunctionBay, Inc. is subjected to the copyright laws of the Republic of Korea and other countries and is provided under a license agreement that restricts copying, disclosure, and use of such documentation. FunctionBay, Inc. hereby grants to the licensed user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the FunctionBay, Inc. copyright notice and any other proprietary notice provided by FunctionBay, Inc. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of FunctionBay, Inc. and no authorization is granted to make copies for such purpose.

Information described herein is furnished for general information only, is subjected to change without notice, and should not be construed as a warranty or commitment by FunctionBay, Inc. FunctionBay, Inc. assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the Republic of Korea and other countries. UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

Registered Trademarks of FunctionBay, Inc. or Subsidiary

RecurDyn is a registered trademark of FunctionBay, Inc.

RecurDyn/Professional, RecurDyn/ProcessNet, RecurDyn/Acoustics, RecurDyn/AutoDesign, RecurDyn/Bearing, RecurDyn/Belt, RecurDyn/Chain, RecurDyn/CoLink, RecurDyn/Control, RecurDyn/Crank, RecurDyn/Durability, RecurDyn/EHD, RecurDyn/Engine, RecurDyn/eTemplate, RecurDyn/FFlex, RecurDyn/Gear, RecurDyn/DriveTrain, RecurDyn/HAT, RecurDyn/Linear, RecurDyn/Mesher, RecurDyn/MTT2D, RecurDyn/MTT3D, RecurDyn/Particleworks I/F, RecurDyn/Piston, RecurDyn/R2R2D, RecurDyn/RFlex, RecurDyn/RFlexGen, RecurDyn/SPI, RecurDyn/Spring, RecurDyn/TimingChain, RecurDyn/Tire, RecurDyn/Track_HM, RecurDyn/Track_LM, RecurDyn/TSG, RecurDyn/Valve

are trademarks of FunctionBay, Inc.

Edition Note

This document describes the release information of **RecurDyn V9R4**.

Table of Contents

Introduction 4
Task Objectives4
Requirements5
Tasks5
Estimated Time to Complete5
Calling the Inital Model 6
Task Objective6
Estimated Time to Complete6
Calling the Rdyn model7
Running the Initial Simulation on the 4-Cylinder Engine Model
Creating an RFlex Body 9
Task Objective9
Estimated Time to Complete9
Creating an RFlex Body 10
Conducting the Dynamic Analysis on the RFlex Body and Reviewing the Results12
Conducting the Durability Analysis17
Task Objective
Estimated Time to Complete17
Conducting the Durability Analysis18
Analyzing and Reviewing the Results
Task Objective
Estimated Time to Complete
Analyzing the Safety Factor Results



Introduction

Fatigue and durability analyses are designed to determine how long a flexible body or specific area of a flexible body modeled in RecurDyn can stably endure various dynamic loads. Such analyses can also determine how stable a body is. The focus on time distinguishes these forms of analysis from other analysis methods, such as those used to determine maximum stress and maximum deformation rates.

RecurDyn, in consideration of the flexibility of the model, supports RFlex of FFlex bodies in multi-body dynamic models. Therefore, this tutorial teaches you how to use the RecurDyn/Durability module to analyze the durability of both the FFlex bodies and the RFlex bodies.

The model being used in this tutorial is a simplified 4-cylinder internal combustion engine. The crankshaft in this model has been replaced with an RFlex body, and the combustive explosion process occurring in the four cylinders push the pistons to provide a dynamic load. The durability analysis in this tutorial determines the stability and durability of the crankshaft design.

Task Objectives

This tutorial covers the following:

- Replacing a flexible body using RecurDyn/RFLEX
- Verifying stresses using RecurDyn/RFLEX
- Recognizing the requirements for the durability analysis
- Obtaining durability analysis results
- Analyzing durability analysis results

Requirements

This tutorial is intended for intermediate users who have read and understood the basic tutorial as well as the FFlex and RFlex tutorials available from RecurDyn. If you have not completed these tutorials, then you are advised to complete them before proceeding with this tutorial. In addition, this tutorial requires a basic understanding of dynamics and the finite element method.

Tasks

This tutorial is composed of the following procedures. This table also shows the time required to complete each task.

Procedures	Time (minutes)
Calling an Rdyn model	10
Replacing an RFlex body.	15
Creating a patch set to verify the fatigue results	5
Conducting a fatigue evaluation	25
Verifying the fatigue results	10
Total	65



This tutorial takes approximately 65 minutes to complete.



Calling the Inital Model

Task Objective

This chapter teaches you how to open the initial model, simulate it, and observe how a 4-cylinder engine model operates.



10 minutes

Calling the Rdyn model

To run RecurDyn and call the initial model:

- 1. On the desktop, double-click the **RecurDyn** icon.
- 2. When the Start RecurDyn dialog window appears, close it.
- 3. From the File menu, click Open.
- Under the Durability tutorial path, select the RD_Durability_4Cyl_Engine_Start.rdyn file. (The file location: <Install Dir> \Help \Tutorial \Durability \RFlexCrankshaft, ask your instructor for the location of the directory if you cannot find it).
- 5. Click Open.
- 6. The model is shown below opens.



The above figure shows a model of an inline 4-cylinder engine, which consists of a cylinder block, pistons, connecting rods, and a crankshaft. In an actual combustion engine, a gas explosion forces the four pistons vertically into the cylinder block, causing the connecting rods on each piston to rotate the crankshaft. In order to simulate such a process in RecurDyn, you must simulate the timing of the gas explosion in the force profile and directly assign it to the piston bodies as a vibration force.

To save the initial model:

1. From the **File** menu, click **Save As**.

(Save this model in the different path because it is impossible to simulate directly in the tutorial path.)

Running the Initial Simulation on the 4-Cylinder Engine Model

In this task, you will run an initial simulation on the model to understand how it operates.



►

To run the initial simulation:

- 1. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**.
- 2. The Dynamic/Kinematic Analysis dialog window appears.
- 3. Verify the simulation conditions and click the **Simulation** button.

Viewing the results:

When your click the **Play** button from the **Animation Control** group in the **Analysis** tab, the fuel explodes in all four pistons in the following order:

Piston_1 \Rightarrow Piston_3 \Rightarrow Piston_4 \Rightarrow Piston_2. You can verify the size of the arrows indicating the force in the animation. Generally, four processes occur in each stroke of the piston (intake \Rightarrow compression \Rightarrow explosion \Rightarrow exhaust). However, in the dynamic model used in this tutorial, only the gas explosion force generated by explosion is significant. Thus, the force profile was created with respect to the timing of the explosive force, and it was modeled in order to assign force to each piston in the appropriate order.





Creating an RFlex Body

This chapter teaches you how to conduct a fatigue analysis on Flexible Bodies.

Task Objective

This task teaches you how to replace an existing rigid body with a flexible body using the RFlex body feature provided in RecurDyn/RFlex and conduct a fatigue analysis on the flexible body.



15 minutes

Creating an RFlex Body

To create the RFlex body:

ImportRFI

- 1. From the **RFlex** group in the **Flexible** tab, click **Import RFI**.
- 2. Change the modeling option to **Body**.



- 3. In the **Working window**, select the **Crankshaft**, as shown below.
- 4. The RFlex Body Import dialog window appears.



- 5. In the **RFlex Body Import** dialog window, perform the following:
 - Click the "..." button to the right of the **RFI File Name** text box.
 - Select the Crankshaft_RFlex.rfi file, which is located in the same folder as the RD_Durability_4Cyl_Engine_Start.rdyn file.
 - Click the "..." button to the right of the Reference text box.
 - In the Database pane, under the Ground group, drag the RFlex_Reference marker and drop it in the Navigation Target window, as shown in the figure to the right.

- ×	Database $ abla 4 \times 10^{-10}$
Navigation Targ ×	★ 4CylinderEngine
Drop have from database I	Groups
Drop here from database :	i⊒
	🖨 🗤 🗊 Ground
	🖃 🛵 Markers
	歳 InertiaMarker
	🕰 Marker2
	RFlex_Reference
	i ⊡ i Connect_Rod_1

- 6. Verify that the most RFlex Body Import recently selected conditions match those RecurDyn RFlex File shown in the figure to the lity_RFlex_Crankshaft\Crankshaft_RFlex.rfi **RFI File Name** right, and then click **OK**. Crankshaft Body(Swapped) Ground.RFlex_Reference Reference OK Cancel
- 7. The **RFlex body** will replace the **Crankshaft Body**.
- 8. Click **Icon Control** in the toolbar, select All Icons and view the results. Ensure that all of the joints that are applied to the existing crankshaft are still applied to the RFlex body. After verifying the joints, clear the selected icons.



Conducting the Dynamic Analysis on the RFlex Body and Reviewing the Results

To conduct a dynamic analysis on the RFlex body:

- 1. Select and right-click **Crankshaft** and click **Properties**.
- 2. The **Properties of RFlexBody1** dialog window appears.
- 3. In the **Properties of RFlexBody1** dialog window, on the **Body** tab, click **Initial Velocity**.
- 4. In the **Body Initial Velocity** dialog window, perform the following:
 - In the **Rotational Velocity** group, select the **X** checkbox, and click **PV**.
 - Select **Initial_Velocity** in the **Parametric Value List** dialog window.
 - Click **M** to the right of the **Reference Marker** text box.
 - In the Database window, drag the Ground Inertia Marker and drop it in the Navigation Target window. Click Close to close the dialog window.
- 5. The entire procedure is shown below.

	Properties of C	Crankshaft [Curr	ent Unit : N/kg	/mm/s/deg]		
	General	Graphic Prog	oerty Or	igin & Orientation		
	Body	FEInfo.	RFlex	Node Scope		
	Marc 17.27	16029642466				
	Ivy 2020	10330043400	IV/ 2.5042			
	3038	1.2256508931	3.5817	4021017135e-002		
Body Initial Velocity	×	8.657997228	lyz 4.5783	2287429483e-002		
Translational Velocity		9.062440403	Izx 0.2896	30668979953		
□ x 0.	Pv	ass Center 0.37	710361874022, -1	15.741374278685, 22]	
□ Y 0.	Pv		Initial \	/elocity	1	
z 0.	Pv	Parame	tric Value List			
Beference Marker		Parame	tric Values			
		No	DP	Name	Value	Comment
Rotational Velocity		1	RPM		3000. E	
□ X 0.		2	Degree		720. E	
Y 0.	Pv	3	End_Time		EX_End_Time E	
		4	Initial_Vel	ocity	Ex_Velocity E	
Z 0.	Pv					
Reference Marker	M					
Close	Cancel					
	Scope	1.00		Apply Apply		
Body Initial Velocity	^	_		• ×	Database	үң х
Translational Velocity					\star 4CylinderEnd	aine
L x 0.	Pv	Nav	rigation Tai	rg 🗖 👗 📗	A Groups	,
□ Y 0.	Pv	Droi	o here from	database !		
□ z 0.	Pv				E-Bodies	
Reference Marker	M				Grou	nd
Rotational Velocity					⊟ <u>¢</u>	larkers
X Initial_Velocity	Pv				δ	InertiaMarker
□ Y 0.	Pv					Marker2
□ z 0.	Pv					RFlex_Reference
Reference Marker Ground.InertiaMarker	м				🕀 👘 Conn	ect_Rod_1
Close Can	cei					



- 6. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**. When the dialog window appears, click **Simulation** to run the analysis without changing the settings.
- 7. It does not take long to complete the simulation. During the simulation, the animation is similar to the previous crankshaft body animation.



To display the distribution of stress in the RFlex body:

- 1. From the **RFlex** group in the **Flexible** tab, click **Stress Strain Shape Generation**.
- 2. The Stress Shape Generation dialog window appears.
- 3. In the **Stress Shape Generation** dialog window, perform the following:
 - For the **RecurDyn/Flex Input File**, specify the RFI file.
 - Select Stress Shape.
 - Click Generate.

Strain Stress Shape Gene	ration
BacurDup (Elay Input Eila	03\Crapkchaft_PElev_rfi
Recurbyn/riex input rife	
	Strain Snape
	✓ Stress Shape
Generate	Close

4. The status dialog window appears to display the stress shape creation progress, as shown in the figure on the right.

Allocating memory Computing strain & stress shape. Writing strain or stress shape in RFI file. Deleting memory. Finished successfully. Elapsed Time = 0 hour : 0 min: 3 sed	
	Close

5. After the stress shape is created, click **Close** in both dialog windows.

Tip: Since the RFI file provided in this tutorial includes only the mode shape information, you must add the stress shape information to the existing RFI file in order to view the stress result. Naturally, this process will increase the size of the RFI file.

To view the simulated stress results in the contour view of **RecurDyn/RFlex**, you must create output files, *.srd files, in the output folder. These files save the stress results for all the nodes in the RFlex body. You can still see the results if you do not create output files, but it may slow the contour animation.





- 6. Under **Contour**, click Output Regenerator.
- 7. The **Output File Generator** dialog window appears, as shown in the figure to the right.
- 8. Click **Output File Setting**.

Out	put File Re	gene	rator								×
Set :	same option	s for	all RFlex bod	lies							
Di	splacement I	Data	(*.rfa) 🖂	Strain /	Stress	Data (*.	erd, *.srd)				
She	II Recovery T	ype fo p	or Contour -	OBott	tom		Displacem	ent Precision () Float	•.rfa)	ouble	
Bea	m Recovery 1 Max Dist	lype f ance	or Contour	⊖¢	С	D	() E	⊖ F	O Max Von	Mises Stress	
N	Bodies Crankshaft	f	Displace	erd,	Stra	Stre	Strain Shell T	Stress Shell T	Beam Type	Dis. Preci	
Use	Animation C	onfiç	guration		_	_	_				
Strain	/ Stress Out	put f	File Setting					Generate	OK	Cance	

- 9. The **Output File Setting** dialog appears as shown in the figure on the right.
- 10. In the Stress group, click Select All.
- 11. Click Close.

Output File S	etting		
Strain —			
EX	EY	EZ	
EXY	EYZ	EZX	
E1	E2	E3	Clear All
EINT	EMISES		Select All
- Stress			
SX 💟	SY 🗸	SZ 🗸	
SXY	SYZ	SZX	
<mark>√ S</mark> 1	✓ S2	<mark>∕∕</mark> S3	Clear All
SINT	SMISES		Select All
Data Precisio	on ————		
🔿 Single (Fl	oat) 🤅	Double	
	Clos	e	

Tip: When creating an output file, only the Von-Mises, Sx, Sy, and Sz tests are included to reduce the file size. If you would like to view other results in the contour, you must select every check button.

12. In the **Output File Generator** dialog window, click **Generate**.

13. When the file is generated, the Stress column in the Information table changes from **Empty** to **Full**. (**Tip:** If the output file was created using the default settings, then the Stress column changes to Partial rather than Full. This is because out of 11 stress tests, only Von-Mises, Sx, Sy, and Sz components are conducted.)

Making animation file	
Current Index / Total Count	1/2 Skip
Animation file for Crankshaft@4Cylin	nderEngine
	Cancel



- 14. In the **RFlex** ribbon, click **Contour**.
- 15. The **Contour** dialog window appears, as shown below.

ontour Option -		Band Option			-View /	Reference N	lode / Refere	nce Mark	er —
nimation Status	SMISES	Legend Type	Display	-	Sel	Body	Node ID	Sel	Ori
/pe	Stress	Location	Bottom	-		Crankshaft	3		
omponent	SMISES -		Show Text Leg	end					
Display Vector	50.7087119001577	Band Level(10~50)		10					
Uniform	Simple	- Style Option							
Contact Surface	Only	Color Option	Edit		- Conto	ur Data Trace	·		
User defin	ed contact surface	Colors	Spectrum	-	Sel	Body	Node	ID	
Contact pa	itches only	Style	Stepped	_					
fin/Max Option —		Text Color	Text Color	•					
/pe User Defined	ı –	Exceed Max Color	Max Color	*					
	Calculation	Less than Min Color	Min Color	-		Add		Delete	
Calc. Result	User Defined -				Conto	ur Element S	et Selection		
1in	0				Sel	Body	Contour	Part	
1ax 72.5	075 2d								
Show Min/Max	Enable Log Scale								
User Defined Ma	ax Color		1						
User Defined Mi	n Color	Mesh Lines	Line Color	•		Add		Delete	

16. In the **Contour** dialog window, perform the following:

- In the **Min/Max Option** group, click **Calculation**.
- In the **Min/Max Option** group, set the **Type** to **User Defined**.
- In the Max text box, type 20.
- Click **OK** to close the dialog window.

17. Click Animation Play.

►

18. The results appear on the model, as shown in the right figure.





Conducting the Durability Analysis

Task Objective

This chapter teaches you how to analyze the durability of the RFlex body.



30 minutes

Conducting the Durability Analysis

To create a patch set:

- 1. Double-click Crankshaft to enter RFlex Body Edit Mode.
- 2. From the Set group in the RFlex Edit tab, click Patch Set.
- 3. Click the **Add/Remove** button, as shown below. Then, hold down the left mouse button and drag the cursor across the model to select the entire body.



- 4. Right-click the selected body and click **Finish Operation** in the context menu.
- 5. In the Patch Set dialog window, click **OK**.
- 6. After the patch set has been created, click **Exit** in the Exit group of the **RFlex Edit** tab to return to the parent mode.





To retrieve the animation file:

From the **Animation Control** group in the **Analysis** tab, click **Reload the last animation file**, as demonstrated below.

	<u>)</u> 📙 🛛	🖻 📶 🥱 •	🦻 - 🍕 🤞	-							
	Home	SubEntity	Analysis	Professiona	l Flexible	Dur	ability	TSG	CoLink	AutoDesign	Co
	\checkmark						[4 44 _i) • 🚳 🗄	.
Dyn/Kir	n Eigen	Scenario	DOE	Pause	Resume Sto	р	昍듲Ы	2	0 Init		
Ť	Simul	ation Type		Simul	ation Control				Animation Co	ntrol	

Note that all of the animation-related buttons are activated.

Tip: When creating the patch set for the RFlexBody1, it may appear as though the previous analysis results are no longer available. However, that procedure does not affect the dynamic analysis results. Therefore, there is no need to perform the dynamic analysis again. You can simply retrieve the animation file, or the RAD file, for the previously analyzed result.

To set the analysis preferences:



- 1. From the **Durability** group in the **Post Analysis** tab, click **Preference** to open the Preference dialog window.
- 2. In the Preference dialog window, on the **Material** tab, specify the path to the **Material Library file** for the fatigue analysis.

(C:\Users\<Your Windows Login ID>\Documents\RecurDyn\<RecurDyn Version> or an equivalent path depending on the OS environment)

3. On the Fatigue Influencing Factors tab, in the Fatigue Factors group, set the Notch Factor Amp (Kf, Kt) to 1.2, as shown below.

Convergence Control	Rainflow Cou	inting
Material	Fatigue Influencing Fa	ctors
- Fatigue Factors		
Notch Factor Amp (Kf, Kt)	1.2	Pv
Surface Factor (ms)	Polished 🔻 1.0	
Size Factor (md)	1.	Pv
Load Factor (mt)	1.	Pv
Other Factor (mo)	1.	Pv
Overall Scale Factor		
Scale Factor (fs)	1.	Pv

- 4. The **Notch Factor** increases the analytically derived stress value to account for stress concentrations due to cracks, holes, and notches (V grooves) caused by the design and processing of the structure. Therefore, the larger the notch factor value is, the more severe the durability analysis will be.
- 5. In the Preference dialog window, do not change the Convergence Control or Rainflow Counting values, and click **OK**.

To conduct the fatigue evaluation:



- 1. From the **Durability** group in the **Post Analysis** group, click **Fatigue Evaluation**.
- 2. The **Fatigue Evaluation** dialog window appears.
- 3. In the **Fatigue Evaluation dialog** window, perform the following:
 - For the **Axial Mode**, select **Bi-Axial**.
 - Change the Life Criteria to Safety Factor.
 - For the Life Criterion, select Goodman.
 - Set the **Searching Increment** to **5 Deg**.

Fatigue Evaluation Axial Mode Life Criteria	O Uni-Axial		Bi-Axial	×
O Stress - Based	🔿 Strain -	Based	Sa	afety Factor
Life Criterion		Soodman		-
Mean Stress Effect		Soodman		Ψ
BWI Weld		lass B		Ψ
Num of Std.Deviations	2	2.		
Searching Increment	1	i Deg		•
Material				
Material < mm-N >	[Steel] 1020 [9	Sample.xml]	н
Element / Patch Set				EL
Time History		History_1		SEL
Occurrence	1	6.		
Pre-Stress File				
Recalculate Recovery Data				
- Fatigue Results				
Time Range	Face No	de ID	Safety Fac	tor (Min.)
Fatigue Tools Import	G	alculation	ОК	Cancel

- 4. In the **Materials** group, click the "…" button.
- 5. When the **Material Manager** dialog window appears, perform the following:
- 6. Select and right-click **[Steel] 1020**, and click **Make Active** in the context menu, as shown in the figure to the right.
- 7. Click OK.

elect	Sample.xm	I	-	New Lib	rary
			Unit	mm-N	•
Name 🛆		Reference	Descript	tion	Yi
[Steel] H11/660		SAE J1099, FEB. 1975	Entry 8, data	ref 6/[
[Steel] RQC100		SAE J1099, FEB. 1975	Entry 12, dat	ta ref 1	=
[Steel] 10B62		SAE J1099, FEB. 1975	Entry 13, dat	a ref 7;	
[Steel] 1005/90		SAE J1099, FEB. 1975	Entry 17, dat	a ref 7	
[Steel] 1005/327		SAE J1099, FEB. 1975	Entry 16, dat	a ref 7	
[Steel] 1015/80		SAE J1099, FEB. 1975	Entry 18, dat	ta ref 4	
[Steel] 1045/225		SAE J1099, FEB. 1975	Entry 21, data ref 7		
[Steel] 1045/390		SAE J1099, FEB. 1975	Entry 23, data ref 7		
[Steel] 1045/410		SAE J1099, FEB. 1975	Entry 22, data ref 7		
[Steel] 1045/450		SAE J1099, FEB. 1975	Entry 24, dat	ta ref 7	
[Steel] 1045/500		SAE J1099, FEB. 1975	Entry 25, dat	ta ref 7	
[Steel] 1045/595		SAE J1099, FEB. 1975	Entry 26, dat	ta ref 7	_
[Steel] 1020		CAR 14000 EED 4075	Entry 19, dat	ta ref 1	
[Steel] 1040	Ma	ke Active	Entry 20, dat	ta ref 1	
[Steel] 1144	Edi	t	Entry 32, dat	ta ref 1	
[Steel] 1541F	Nev	v Material	Entry 34, dat	a ref 1	
[Steel] 4130/258			Entry 39, dat	ta ref 1	
[Steel] 4130/365	Cut		Entry 40, dat	ta ref 1	
[Steel] 4142/380	Cop	у	Entry 44, dat	ta ref 6	
[Steel] 4142/400	Pac	ta	Entry 45, dat	ta ref 6	
ISteel 4142/450	FOS	ic I	Entry 46 dat	a raf 6	- ¹
•	Del	ete 🛛			

- 8. Click the **EL** button to the right of the **Element/Patch Set** text box.
- 9. Select the Patch Set for RFlexBody1.
- **10.** A Time History set is defined already, in the **Time History** dialog window. To change the range of time, click **R**.

T	ime ⊦	listory	1			
	No	Use	Name	Time Range		Add Row
	1	 Image: A start of the start of	History_1	1,721	R	Insert Row
						Delete Row
						Clear
					1	
				OK Cancel		

- 11. In the **Time Range** dialog window, click **All**.
- 12. Click **Close** in the **Time Range** dialog window.
- 13. Click **OK**.

Use	Frame	Time Step	Ľ
\checkmark	1	0.	
\checkmark	2	5.55555555555	
\checkmark	3	1.11111111111	
\checkmark	4	1.66666666666	
\checkmark	5	2.2222222222	
\checkmark	6	2.77777777777	
\checkmark	7	3.33333333333	
\checkmark	8	3.8888888888	
\checkmark	9	4.4444444444	
\checkmark	10	5.e-004	
\checkmark	11	5.55555555555	

14. Click Calculation.

fime Range	Face Node ID	Safety Factor (Min.)	
History_1	77,1909,1910	2.18658827852219	

15. A progress bar appears to display the fatigue analysis progress. Upon completion of the analysis, the results will appear in the results group, as shown in the figure to the right.

- 16. In the **Fatigue Evaluation** dialog window, click **Fatigue Tools**.
 - Click the **Rainflow Counting** button in the Fatigue Tools dialog.

As shown below, the **Rainflow Counting** results in the Excel are based on the Stress Time History applied to the patch zone where the damage is largest. The results are displayed in the numbers of cycles according to the stress amplitude and the mean stress.

Fatigue Tools
Time History History_1
Patch Info.
Max Damage OUser-defined Sel
Node IDs 77,1909,1910
Rainflow Counting Plot History
Close



Click the **Plot History** in this dialog.

As shown below, it is possible to check the **Stress Time History** on the patch which has the maximum damage results among the defined patches.



To verify the contour results:

Contour

- 1. From the **Durability** group in the **Post Analysis** group, click **Contour**.
- 2. The **Durability Contour** dialog window appears.
- 3. In the **Durability Contour dialog** window, perform the following:
 - Click **Calculation**.
 - Select Enable Log Scale.
 - Click **Contour View to** view the results.

Durability Contour	
Time History	History_1
View Type	Contour 🔻
Contour Option	Style Option
Safety Factor	Color Option Edit
Band Option	Colors Spectrum 🔻
Legend Type Display 🔻	Style Stepped 💌
Location Bottom	Text Color 📃 🔻
Show Text Legend Band Level(10~50) 10	
- Min/Max Option	Mesh Lines
Display 🔻	Vector Color
Calculation	Vector Size Factor 5.07
Min 2.7006 2.7006	Probe Option
Max 22918 22918	O Node Select
Show Min/Max	Patch Clear
Enable Log Scale	Show Probe Results
Export	View OK

4. To make the results more visible, click **Edit** in the **Style Option** group of the Durability <u>Contour dialog window, and change the colors as follows.</u>

Select Band Color	Select Band Color
First Band Color (Min) • Last Band Color (Max) • OK Cancel	First Band Color (Min) Last Band Color (Max)

5. Click **Contour View** again to highlight the least durable sections in red, as shown below. This makes it easier to identify the areas with a relatively short fatigue life. (**Tip:** At this point, if you would like a more detailed Contour Plot, then select **Wireframe** in the toolbar before viewing the results.)



To change the materials and conduct another fatigue evaluation:

- In the Fatigue Evaluation dialog window, in the Material group, click the "..." button.
- 2. The **Material Manager** dialog window appears.
- 3. Right-click the list, and click **New Material** in the context menu, as shown in the figure to the right.

Material Manager		
Select Sample	.xml	▼ New Library
		Unit mm-N 💌
Name	Reference	Description Yi
[Steel] MANTEN	SAE J1099, FEB. 1975	Entry 61, data ref 3 😑
[Stainless Steel] 304/160	SAE J1099, FEB. 1975	Entry 36, data ref 1
[Stainless Steel] 304/327	SAE J1099, FEB. 1975	Entry 37, data ref 1
[Stainless Steel] 310/145	SAE J1099, FEB. 1975	Entry 38, data ref 1
[Steel] AM350	SAE J1099, FEB. 1975	Entry 5, data ref 1/[
[Steel] GAINEX	SAE J1099, FEB. 1975	Entry 7, data ref 7/[
[Steel] H11/660	SAE J1099, FEB. 1975	Entry 8, data ref 6/[
[Steel] RQC100	SAE J1099, FEB.	Make Active
[Steel] 10B62	SAE J1099, FEB.	Edit
[Steel] 1005/90	SAE J1099, FEB.	
[Steel] 1005/327	SAE J1099, FEB.	New Material
[Steel] 1015/80	SAE J1099, FEB.	Cut
[Steel] 1045/225	SAE J1099, FEB.	Canu
[Steel] 1045/390	SAE J1099, FEB.	Сору
[Steel] 1045/410	SAE J1099, FEB.	Paste
[Steel] 1045/450	SAE J1099, FEB.	Delete
[Steel] 1045/500	SAE J1099, FEB.	citity 25, data ici riii
[Steel] 1045/595	SAE J1099, FEB. 1975	Entry 26, data ref 7
[Steel] 1020	SAE J1099, FEB. 1975	Entry 19, data ref 1
[Steel] 1040	SAE J1099, FEB. 1975	Entry 20, data ref 1
	CAE 11000 FER 1075	Entry 32 data ref 1
	ОК	

- 4. In the dialog window to create the new material, perform the following:
 - For the **FatigueLimitStress** value, enter **400**.
 - For the **Name**, enter **Crankshaft_Material**.
 - For the **UltimateStrength**, enter **850**.
 - Click **OK**.

1			∄≣ ⊉↓		
TransitionFatigueLife			 Cyclic Properties		
Misc			 CyclicStrainHardeningExponer	1	
Description			 CyclicStrengthCoefficient		
LastModifiedDate	2017-05-17 16:05:07		 CyclicYieldStrength		
Name	Crankshaft_Material		 Fatigue Properties		
Modifier			 FatigueDuctilityCoefficient		≡
Reference			 FatigueDuctilityExponent		
Tags			 FatigueLimitInCycles		
Monotonic Properties			 FatigueLimitStress	400	
ModulusOfElasticity			 FatigueStrengthCoefficient		
ModulusOfShear			 FatigueStrengthExponent		
PercentReductionInArea		=	 TransitionFatigueLife		
PoissonsRatio			 Misc		1
TrueFractureDuctility			 Description		-
TrueFractureStrength			 LastModifiedDate		
UltimateStrength	850		Name	Crankshaft_Material	
YieldStress		T	 Modifier		1
UltimateStrength			UltimateStrength		

Tip: The material property information required to derive the safety factor is the fatigue limit stress and the ultimate strength. However, the fatigue limit stress is not available in the material property information provided by the material library. In such cases, the cyclic yield stress is used instead of the fatigue limit stress information to derive the safety factor. However, when creating a new material to calculate the safety factor in this case, enter the fatigue limit stress directly.

- 5. Return to the **Material Manager** dialog window, right-click the newly created **Crankshaft_Material** and click **Make Active**.
- 6. In the Material Manager dialog window, click OK.

[High-Temperature Alloy] I	SAE	J1099, JUN1998	ENTRY 12; TABL	LE 4B	
[High-Temperature Alloy] I	SAE	J1099, JUN1998	ENTRY 13; TABL	.E 4B	
Crankshaft_Material					
		<u>M</u> ake Activ	e		
		Edit			
	Q	New Materi	ial		

7. In the Fatigue Evaluation dialog window, click **Calculation**.

8. The safety factor is calculated again, as shown below. The fatigue limit stress and the yield stress of the new crankshaft material are much greater than those of [Steel] 1020. Therefore, the minimum safety factor result is larger than that of the previous result.

Axial Mode Life Criteria —————	🔿 Uni-Axial		Bi-Axial
O Stress - Based	⊖ Strain	- Based	Safety Factor
Life Criterion		Goodman	•
Mean Stress Effect		Goodman	-
BWI Weld		class B	-
Num of Std.Deviations		2.	
Searching Increment		5 Deg	•
Material			
Material < mm-N >		Crankshaft_M	aterial [Sample.xml H
Element / Patch Set		Crankshaft.Se	tPatch1 El
Time History		History_1	SEL
Occurrence		1.	
Pre-Stress File			
Recalculate Recovery Da	ıta		
Fatigue Results			
Time Dange	Face N	ode ID	Safety Factor (Min.)
Time Kange	77 100	9 1910	3.32987598066552
History_1	11,150	.,	

To reset the patch set and derive the safety factor again:

In the previous procedure, you derived the safety factor by selecting the patch set for the entire surface of the RFlex body. Consequently, the calculation takes a considerable amount of time. You can decrease the calculation time by only performing the fatigue analysis on specific sections. Choose these sections based on the Von-Mises stress contour results, and set the patch set only for those specific sections. To do so, follow these steps:

- 1. Double-click the **RFlexBody1** to enter **RFlex Body Edit** Mode.
- 2. From the **Set** group in the **RFlex Edit** tab, click **Patch Set**.
- 3. In the **Patch Set** dialog window, perform the following:
 - For the **Tolerance (Degree)**, enter **60**.
 - Click Add/Remove (Continuous), and select an element of interest, as shown below.
 - If the degree difference between the Normal Vectors of the selected patch and neighboring patches are within the range of 60 degrees, then the system automatically selects it as the patch set.

- Right-click the element and click **Finish Operation** in the context menu.
- On the **General** tab, change the name to **Checking_Point_1**.

neral External Patch Set	General Exte	ernal Patch Set	
Color Automatic 👻	Name	Checking_Point1	
Add/Remove	Unit		
	Force	newton	✓ MKS
Add/Remove (Continuous) Tolerance (Degree) 60	Mass	kilogram	▼ MMKS
Check Reverse Direction	Length	millimeter	▼ CGS
Add/Remove (Select Front)	Time	second	▼ IPS
Add (Noce Set)	Angle	degree	▼ FPS
Preview Normal			
Normal Adjust			
Auto Adjust Switch			
Manual Select Target Switch			
No. of Patches 1456			
No. of Patches			
No. of Patches 1456 OK Cancel		ОК	Cancel Appl
No. of Patches		ОК	Cancel Appl
No. of Patches 1456 OK Cancel		ОК	Cancel Appl
No. of Patches 1456 OK Cancel		ОК	Cancel Appl
No. of Patches 1456 OK Cancel		ок	Cancel Appl
No. of Patches 1456 OK Cancel		ОК	Cancel Appl
No. of Patches 1456 OK Cancel		ОК	Cancel Appl
No. of Patches 1456 OK Cancel		ОК	Cancel Appl
No. of Patches 1456 OK Cancel		ОК	Cancel App!
No. of Patches 1456 OK Cancel		OK	Cancel App!
No. of Patches 1456 OK Cancel		ОК	Cancel Appl
No. of Patches 1456 OK Cancel		ОК	Cancel Appl

Repeat step 3 to create three more patch sets, as shown below. At this point, the names of the patch sets should be Checking_Point_2, Checking_Point_3, and Checking_Point_4.



- 5. After you have created the four patch sets, you can view the patch set information in the database, as shown in the figure to the right:
- 6. Confirm that the patch sets were created successfully, and then click the Exit icon from the Exit group in the RFlex Edit tab to return to the parent mode.
- 7. From the Animation Control group in the Analysis tab, click Reload the last animation file.
- 8. From the **Durability** group in the **Durability** tab, click **Fatigue**.
- When the Fatigue Evaluation dialog window appears, change the name of the Element/Patch Set to Crankshaft.Checking_Point_1. Do not change any other settings.
- **10.** Click **Calculation** to produce the calculation results.
- 11. Note that the calculation time is much faster than before.
- 12. Click **OK**.

Axial Mode	O Uni-Axial		Bi-Axial
Life Criteria			
O Stress - Based	◯ Strain	- Based	Safety Factor
Life Criterion		Goodman	~
Mean Stress Effect			v
BWI Weld		class B	*
Num of Std.Deviations		2.	
Searching Increment		5 Deg	•
Material			
Material < mm-N >		Crankshaft_M	aterial [Sample.xml H
Element / Patch Set		Crankshaft.Checking_Point1	
Time History		History_1	SEL
Occurrence	1.		
Fatigue Results			
Time Range	Face N	ode ID	Safety Factor (Min.)
History_1	1552,15	50,256	7.13752291875641

- 13. From the **Durability** group in the **Post Analysis** tab, click **Contour**.
- 14. In the **Contour** dialog window, click **Calculation**

Immeting Immeting Immet	4CylinderEngine Time = Initial Stat	Durability Contour	
Ver vir Considioninini Serie vir straini Device vir		Time History	History_1
Au Image: Control function Bard Option Bard Option <tr< th=""><th></th><th>View Type</th><th></th></tr<>		View Type	
Sinde Factor Sinde Factor Bind Option Lead 10 20 100 Sinde Factor Discretion Lead 10 20 100 Sinde Factor Discretion Lead 10 20 100 MinMax Option Discretion Discretion<		- Contour Ontion	
and Option legend Type telenan lostow cat legend main Level(10-50 Decent Decent <th>K</th> <td>Safety Factor</td> <td>Color Option Edit</td>	K	Safety Factor	Color Option Edit
Belle Outlow Legend Type Legend Type Ball Level(10-50 Min/Max Option Diploy Calculation Min Max 02516 Diploy Calculation Min Max 02516 Diploy Calculation Note Diploy		- Rand Ontion	Colors Spectrum
Legend Type Understand Location Interdition Display Interdition Min/Max Option Interdition Display Interdition Min/Max Option Interdition Display Interdition Min/Max Option Interdition Display Interdition Min/Max Option Deated Interdition Display Deated Display Interdition Display Display		Distan	Style Stepped V
Location Bettom Bhow Ted Legend Band Level(10-50 Min Max Option Display Calculation Min Max Option Calculation Min Max Option Display		Legend Type Display +	Tert Color
Image: Since Text Legend Band Level(10-50 Image: Since Text Legend Image: Since Text		Location Fottom	
Band Level(10-50 0 Imm/Max Option 0 Imm/Max Option 0 Ispir 0 Calculation 0 Mix 82514 22514 O hode 0 O hode		Show Text Legend	
Min/Max Option Uspiky Ispiky Calculation Wath Lines Vector Size Factor Sor None Results Show Min/Max Deport View View OK View OK View OK View OK View OK	1	Band Level(10~50) 10	
Implay Implay Implay Implay Implay Implay Implay Implay Implay Implay Implay Implay Implay Implay Implay Implay <th></th> <th>Min/Max Option</th> <th></th>		Min/Max Option	
Calculation Vector Color Wetor Size Factor 5.07 Probe Option Probe Option Name & 2005 Probe Results Deport View OK 0 Structure Name Accurate Name Accurate 1.29e+02 2.58e+02 1.02e+03 2.07e+03 4.14e+03 8.28e+03	>	Display	
Image: Non-Weilder Size Factor 5.07 Min 8.0086 Probe Option Nax 2831.4 2831.4 Show Min/Max Patch Clear Enable Log Scale Show Probe Results Deport View OK	1	Calculation	Vector Color
Min 8.0065 LOB6 Probe Option Max 8281.4 9 Patch 1 Clear Show Min/Max 9 Patch Clear Show Probe Results Emable Log Scale 9 Show Probe Results 9 Nover 0 K Show Min/Max 9 Patch 0 K 0 K 9 Patch Show Min/Max 9 Patch 0 K 0 K 9 Patch Show Min/Max 9 Patch 0 K 0 K 9 Patch Show Min/Max 9 Patch 0 K 0 K 9 Patch Show Min/Max 9 Patch 0 K 0 K 9 Patch Show Probe Results 9 Patch 0 K 0 K 9 Patch Show Probe Results 9 Patch 0 K 0 K 9 Patch 9 Patch Show Probe Results 9 Patch 0 K 0 K 9 Patch 9 Patch 9 Patch Show Probe Results 9 Patch 9 Patch 9 Patch 9 Patch 9 Patch 9 Patch Show Probe Results 9 Patch 9 Patch 9 Patch <th></th> <td></td> <td>Vector Size Factor 5.07</td>			Vector Size Factor 5.07
Max 8281.4 © Path Clear © Path © Emble Log Scile © Snow Probe Results Export View OK 2 0 3.21e+01 2.43e+01 1.29e+02 2.58e+02 1.03e+03 2.07e+03 4.14e+03 8.28e+03		Min 8.0086 8.0086	Probe Option
Operation Operation <t< td=""><th></th><td>Max 8281.4 8281.4</td><td>ONode Select</td></t<>		Max 8281.4 8281.4	ONode Select
Image: Structure Image: Structure<		Show Min/Max	Clear
Image: sport View OK Subsection 3.21e+01 6.43e+01 1.29e+02 2.58e+02 1.05e+03 2.07e+03 4.14e+03 8.28e+03		Enable Log Scale	Show Probe Results
300+102 1+205+01 3.21e+01 6.43e+01 1.29e+02 2.58e+02 5.16e+02 1.03e+03 2.07e+03 4.14e+03 8.28e+03		Export	View OK VIEW
8.0+e ⁻⁰ 1+60 =+01 3.21e+01 6.43e+01 1.29e+02 2.58e+02 5.16e+02 1.03e+03 2.07e+03 4.14e+03 8.28e+03	N.		
80++07 =+605+01 3.21e+01 5.43e+01 1.29e+02 2.58e+02 5.16e+02 1.03e+03 2.07e+03 4.14e+03 8.28e+03		15	
0.012+00 - 1000+01 - 5,212+01 - 0.452+01 - 1.252+02 - 2.152+02 - 1.102+02 - 1.032+03 - 2.072+03 - 4.142+03 - 6.252+03			
		8.072+00 \$1602+01 3.2	re+U1 6/43e+U1 1/29e+U2 2/36e+U2 3/16e+U2 1/U3e+U3 2/U/e+U3 4/14e+U3 8/28e+U3

- 15. Click **Contour View** to show the result for **Checking_Point_1**, as shown below.
- 16. Perform the same procedure to get the results for Checking_Point_2, Checking_Point_3, and Checking_Point4.



Analyzing and Reviewing the Results

Task Objective

This chapter teaches you how to analyze and review the safety factor results of the model.



10 minutes

Analyzing the Safety Factor Results

The fatigue results from the durability analysis include the safety factor and the fatigue life. In general, a safety factor represents the relationship between the maximum stress and the allowable stress. However, the safety factor in the fatigue analysis indicates a different relationship.

 As shown below, the straight (Goodman method) and parabolic (Gerber method) equations for the relationship between the fatigue limit stress and the ultimate strength produce the minimum safety factor using the Rainflow Counting results (mean stress, stress amplitude) necessary for durability analysis.



- Therefore, the fatigue limit stress and the yield stress are sufficient to derive the safety factor. In this tutorial, you have also used these two data items to derive the safety factor of the crankshaft material created during the new material creation process.
- The first durability analysis performed in this tutorial took approximately 2 minutes to create the patch sets for the entire RFlex body and derive the safety factor (this time may vary depending on your PC specifications). This may seem like a relatively long time, but it can be decreased as follows:
 - a. Study the **Von-Mises stress distribution** in the **RFlex/Contour** and identify the weak sections of the structure.
 - b. Set the patch sets for the weak sections only.
 - c. This method is essential if you need to derive fatigue results repeatedly while varying the durability analysis conditions.





• The following table compares the safety factors derived from the four patch sets for the initially chosen material ([Steel] 1020) and the new crankshaft material.

	Minimum Safety Factor	
Material	Crankshaft material	[Steel] 1020
Patch Set		
Checking_Point_1	7.389	4,300
Checking_Point_2	13.340	8,361
Checking_Point_3	7.896	4.599
Checking_Point_4	3.403	3.329

- Since the proportional limit and the yield stress for the new crankshaft material are greater than those for [Steel] 1020, the new crankshaft material has a higher safety factor. Furthermore, you can see that the section where the smallest safety factor is derived from is Checking_Point_4.
- In general, the Safety Factor in a structure can be expressed by the relationship between the maximum stress and the yield stress. Assuming the maximum stress at

Checking_Point_4 of the crankshaft used in this tutorial is 72 Mpa and the yield stress of [Steel]1020 is 262 Mpa, then the safety factor could simply be expressed in 262/72=3.63. However, the result you got in this tutorial is 2.24, and you can see that there is a difference.

• If the crankshaft does not get dynamic loads in the engine but gets only static loads, then it is significant to derive the safety factor with the yield stress and the maximum stress. However, when it is the structure exhibiting dynamic behaviors such as the crankshaft, the safety factor derived from fatigue analysis does become significant.

Thanks for participating in this tutorial